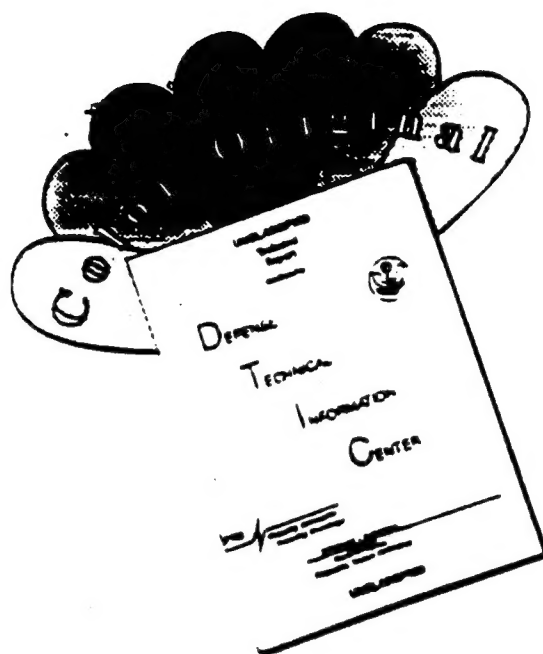


REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-018	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE November 1993		3. REPORT TYPE AND DATES COVERED
4. TITLE AND SUBTITLE SOLVE FLUID FLOW PROBLEMS WITH "PHOENICS"			5. FUNDING NUMBERS	
6. AUTHOR(S) Dr. Cal Kodres and Dr. Gene Cooper				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Naval Facilities Engineering Service Center Port Hueneme, CA 93043-4328			8. PERFORMING ORGANIZATION REPORT NUMBER TDS-2001-ENV	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)			10. SPONSORING/MONITORING AGENCY REPORT NUMBER	
11. SUPPLEMENTARY NOTES				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release; distribution is unlimited.			12b. DISTRIBUTION CODE <input checked="" type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	
13. ABSTRACT (Maximum 200 words) Save time and money by using the computational fluid dynamics program "PHOENICS" for design, optimization, and troubleshooting of fluid flow systems.				
19950112 011			By _____	
			Distribution / _____	
			Availability Codes	
Dist		Avail and/or Special		
A-1				
14. SUBJECT TERMS PHOENICS, computational fluid dynamics (CFD), high velocity, low velocity			15. NUMBER OF PAGES 6	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT UL	

DISCLAIMER NOTICE



THIS DOCUMENT IS BEST QUALITY AVAILABLE. THE COPY FURNISHED TO DTIC CONTAINED A SIGNIFICANT NUMBER OF COLOR PAGES WHICH DO NOT REPRODUCE LEGIBLY ON BLACK AND WHITE MICROFICHE.



Technical Data Sheet



November 1993

Naval Facilities Engineering Service Center
Port Hueneme, California 93043-4328

TDS-2001-ENV

SOLVE FLUID FLOW PROBLEMS WITH "PHOENICS"

Save time and money by using the computational fluid dynamics program "PHOENICS" for design, optimization, and troubleshooting of fluid flow systems.

Computational Fluid Dynamics (CFD) is a computer modeling technique for mathematical analysis of engineering and environmental processes. This technology increases the efficiency and effectiveness of conceptual design and improves the reliability of prototyping. CFD also serves as a practical alternative in areas where physical experiments are prohibitive.

CFD is applicable to nearly all fluid flow problems. Examples of just a few applications include:

- Drag and moments caused by currents past complex structures or ships at anchor.
- Viscous flow around ships or fully submerged appended bodies.
- Turbulent ship airwake prediction for helicopter approach.
- Movement of sediment in harbors.
- Computation of blast wave forces on buildings and the influence that berms, walls, and barriers have on the strength and propagation of blast waves.
- Propagation of fire through a building, or the spread of fire retarding agent.

- Circulation of air in paint hangars, or thermal stratification in hangars, or more generally, the thermodynamic field and circulation of air in any space with a ventilation system.

- Air flow past a building for applications including natural ventilation of buildings, estimation of forces on tents and soft structure buildings, and prediction of vulnerability from chemical, biological, or radiation attack.

- Flow and temperature differences between power station intake and discharge ports in a bay.

- Dispersion of hot water or foreign fluids released into a river or bay.

- Dispersion of particles released from a smoke stack.

For the past decade, the Naval Facilities Engineering Service Center (NFESC) has been using a CFD program called PHOENICS. PHOENICS was developed by Professor D. B. Spalding of the Imperial College, London, and is widely accepted as the leading, general purpose, CFD program in the world.

Solving fluid flow problems with PHOENICS is a three-step process. First, a grid is constructed,

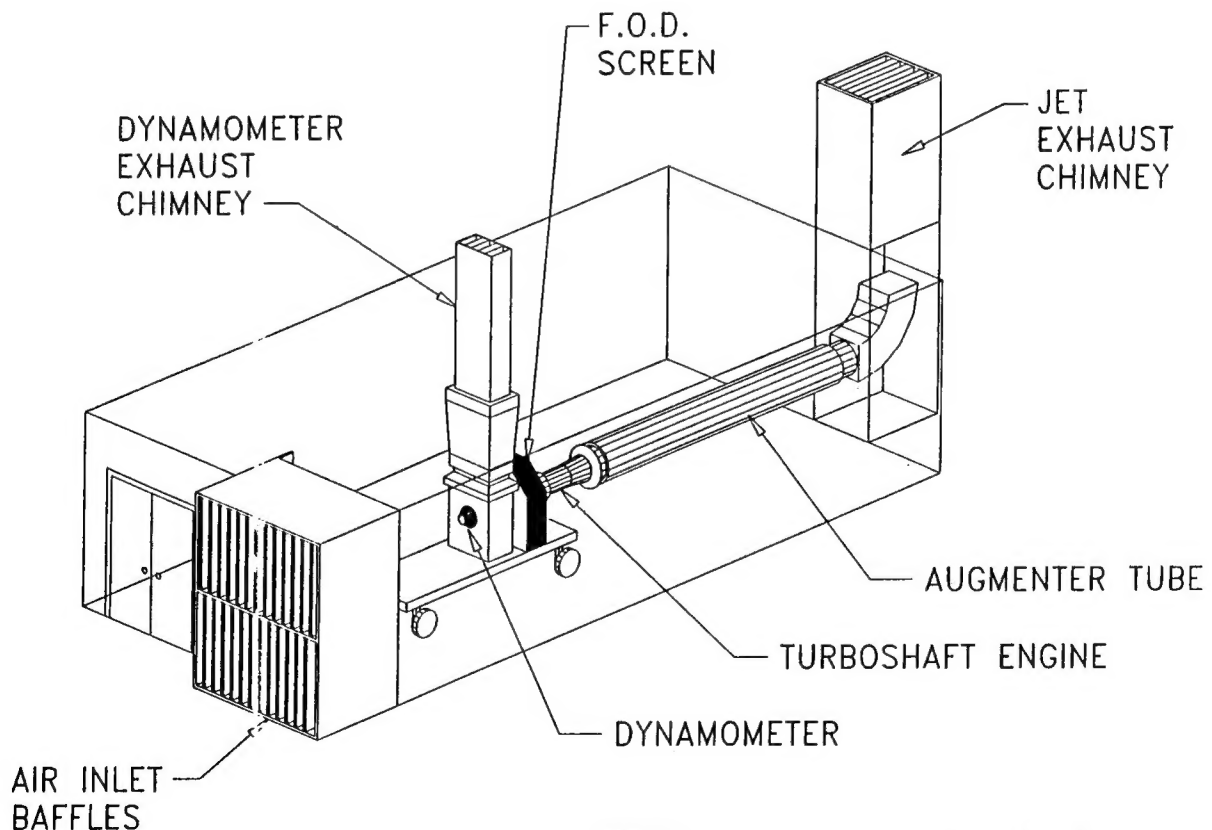
modeling the flow geometry. Second, boundary and initial conditions are stipulated: temperatures, pressures, and/or velocities. Finally, the PHOENICS program is run on a computer to determine all the unknown fluid properties. Once the model has been developed, different scenarios are examined by changing the geometry or changing the boundary conditions and rerunning the PHOENICS program.

PHOENICS is capable of simultaneously analyzing the following flow conditions:

- Transient or steady state flows
- 1-, 2-, or 3-dimensional flows
- Chemically reactive or inert materials; exothermic and endothermic reactions
- Complex geometries and boundary conditions
- Single-, double- or multi-phase flows
- Laminar or turbulent flows
- Incompressible or compressible flow
- Parabolic or elliptic solutions
- Heat transfer (conduction, convection, radiation)

EXAMPLE OF A HIGH VELOCITY PROBLEM

A PHOENICS CFD model was developed by NFESC for evaluating aerothermal performance and for troubleshooting a new generation of turboshaft engine test cells. This is an environment with high velocity flows and high heat transfer rates. The modeling was complicated by a nonsymmetric geometry.

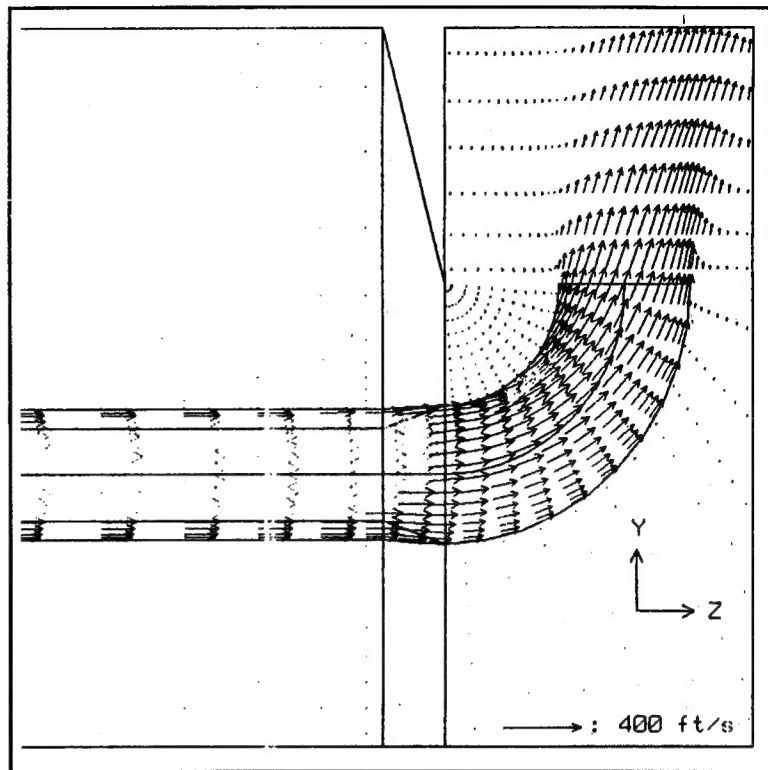


**T700 turboshaft engine test cell at the
Marine Corps Air Facility, Camp Pendleton, California.**

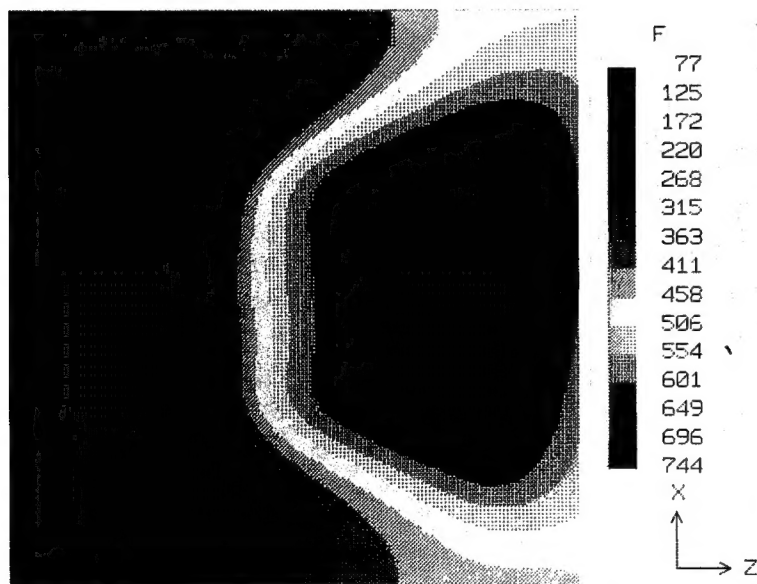
There are many turboshaft engine test cell design problems that are more efficiently analyzed using CFD. For example, the designer can predetermine:

- Velocities and temperatures of engine exhaust gases that are controlled to protect augmentor tube components and to limit noise propagation.
- Velocity and direction of the air entering the engine that must remain within tolerances to achieve accurate simulation of flight performance.

The first opportunity to use the PHOENICS model for troubleshooting occurred during the initial calibration of the test cell. High air temperatures were being measured inside the cell. Spillover of the dynamometer was suspected; some of the dynamometer exhaust was possibly entering the interior of the test cell.

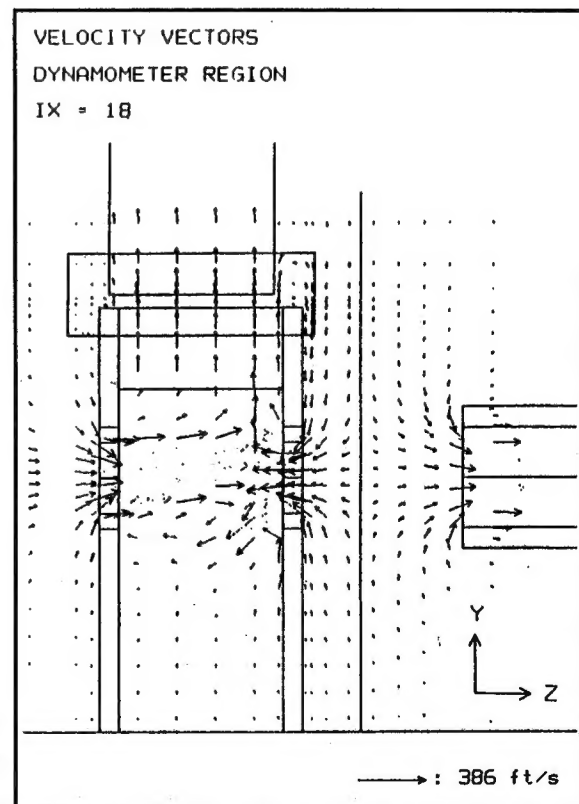


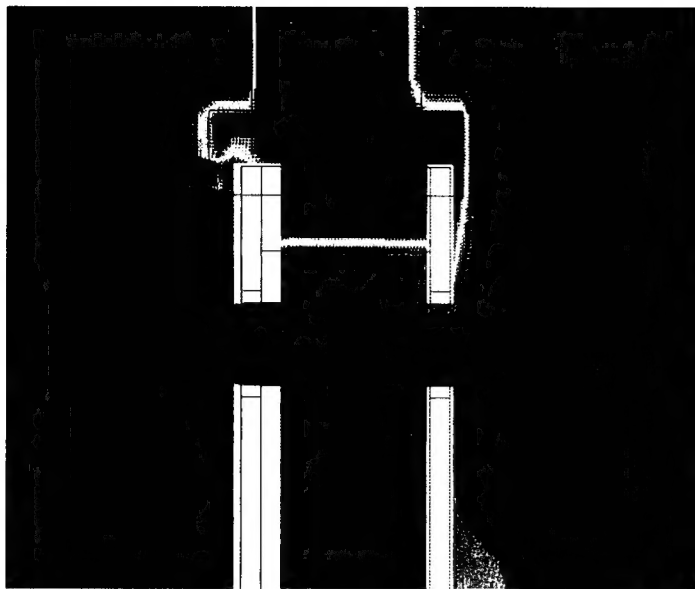
Velocities of the engine exhaust leaving the augmentor tube and passing out the chimney.



Gas temperatures in the exit plane of the chimney.

Vectors showing air velocities and directions at the inlet to the engine. This is a side view that shows both the engine inlet and the dynamometer. Note that the dynamometer and its exhaust stack are not perfectly aligned. Some of the dynamometer exhaust is escaping out of the back. A solid plate is mounted between the dynamometer overflow and reaching the engine inlet.





F
77
101
125
149
173
197
221
245
269
293
317
341
365
389
413



The overflow problem was examined using the PHOENICS model. For this study, alignment of the dynamometer and its exhaust chimney was the major parameter.

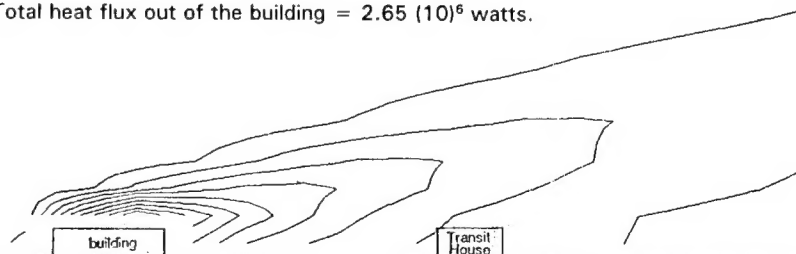
Gas temperatures in the vicinity of the dynamometer. This is the same view shown on the previous figure. The overflow is shown as a high temperature "red" region at the back of the dynamometer.

EXAMPLE OF A LOW VELOCITY PROBLEM

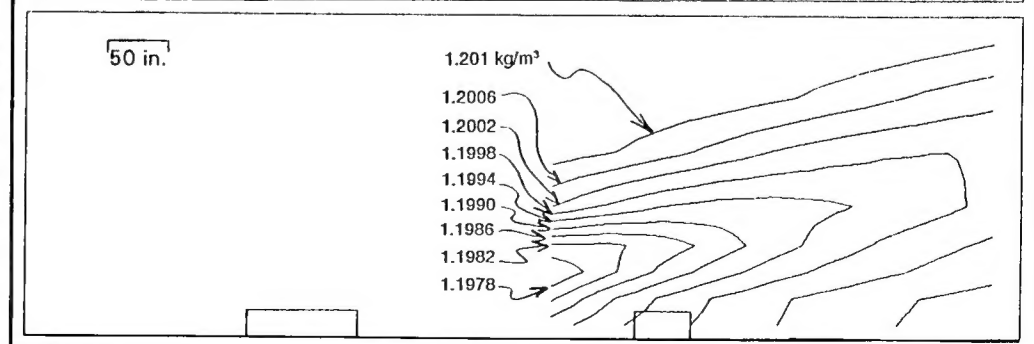
NFESC was asked to calculate density variations in the atmosphere above the Naval Observatory, caused when the wind is heated by the walls of nearby buildings. The Observatory was concerned that refraction of the incoming light was introducing uncorrectable errors into celestial observa-

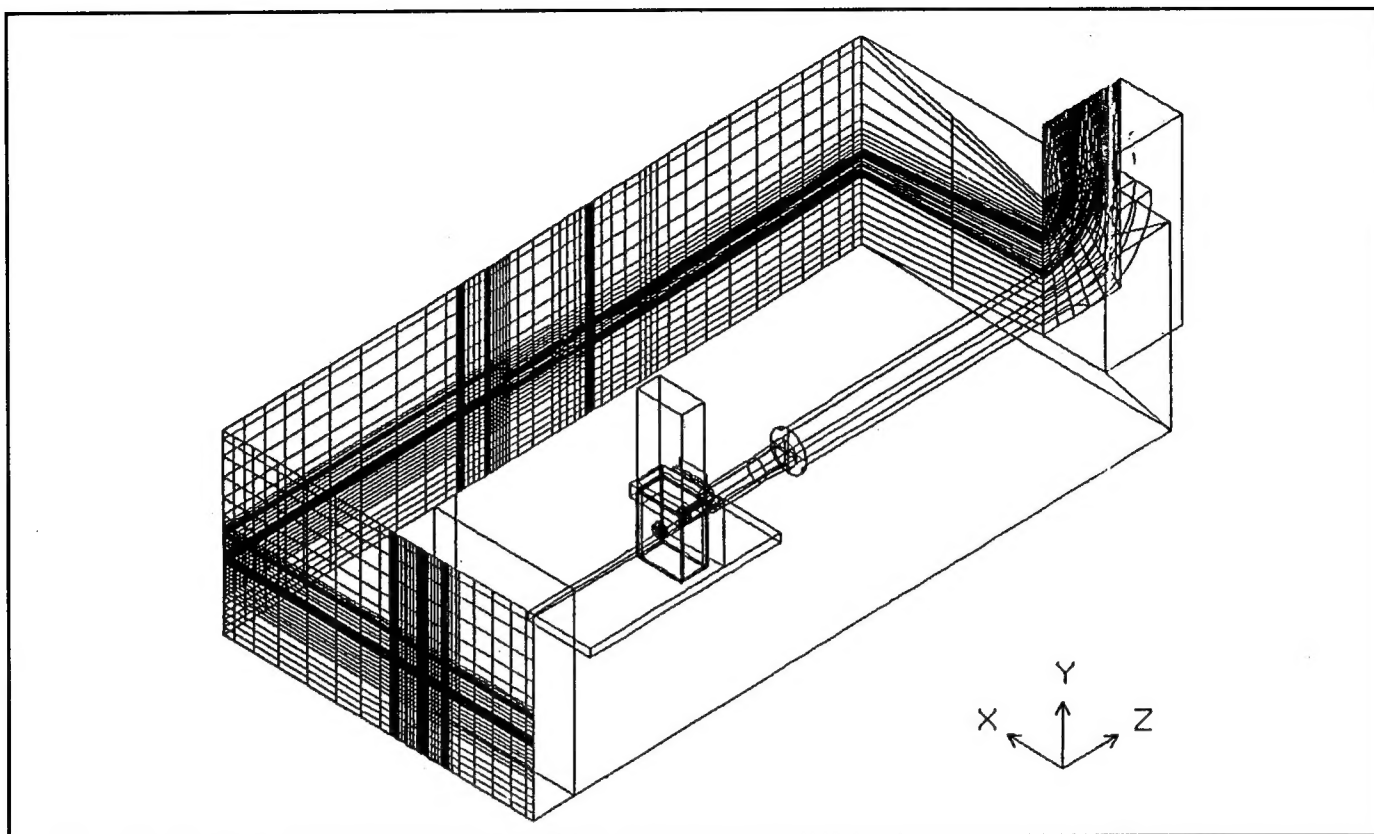
tions. PHOENICS was again used. This problem is the other extreme: very low velocities and very low heat transfer rates. But a very high accuracy was required. The major complication was the modeling of the buoyancy of the heated air.

NOTES: (1) E-W plane for a July evening at 10:00 pm.
(2) Wind at 2 m/sec, 27.2°C, blowing from the west.
(3) Total heat flux out of the building = $2.65 (10)^6$ watts.



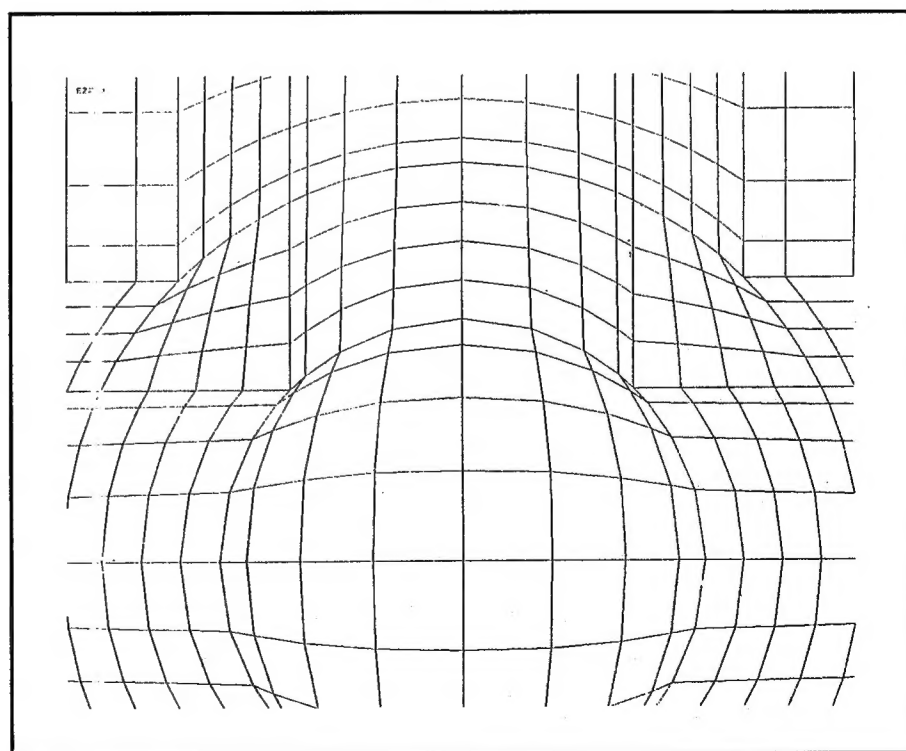
Densities of the air above the Naval Observatory. The wind is blowing past the building (where it is being heated) to the Transit House (where the telescope is located). Note that the hot air is rising as it moves towards the Observatory.



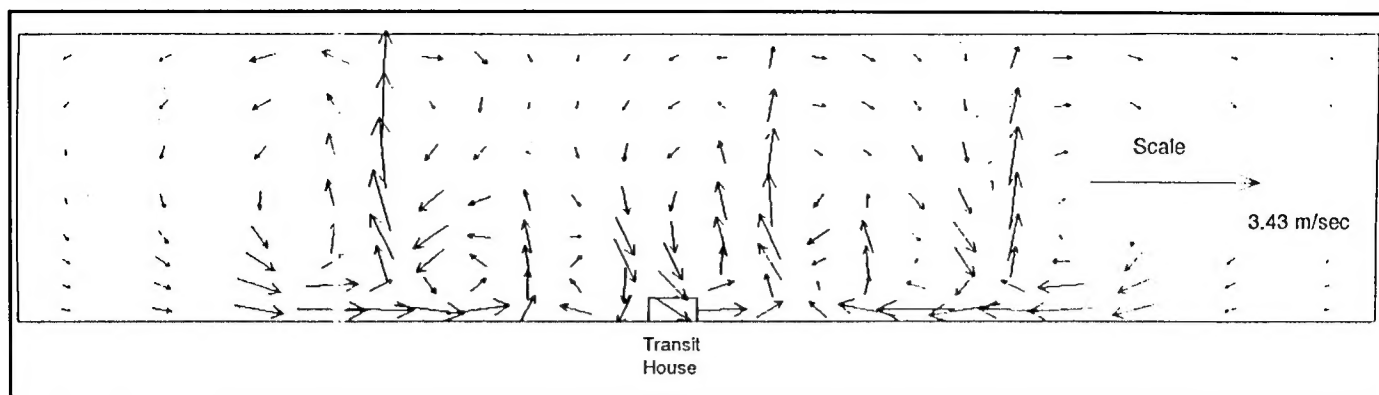


Computational grid constructed to model the Camp Pendleton turboshaft engine test cell.
 (To keep from cluttering this figure, the grid through the interior of the cell is now shown.)

The test cell was approximated using a 40 x 35 x 77 computational grid. The grid was clustered in areas of geometric change, areas of expected high flow gradients, and areas of heat transfer, and was fitted to circular regions such as the nozzle of the engine.



Grid modified to fit the nozzle of the engine.



**Convection currents in the vicinity of the Naval Observatory for a day in July.
This view shows a plane perpendicular to the direction of a 2 m/sec wind.**

For further information on PHOENICS capabilities and
specific applications, contact:

Dr. Cal Kodres

Code ESC421

Naval Facilities Engineering Service Center

DSN: 551-1656; Comm: (805) 982-1656

or

Dr. Gene Cooper

Code ESC64

Naval Facilities Engineering Service Center

DSN: 551-1370; Comm: (805) 982-1370

or

Jerry Dummer

Code ESC101

Naval Facilities Engineering Service Center

DSN: 551-1599; Comm: (805) 982-1599